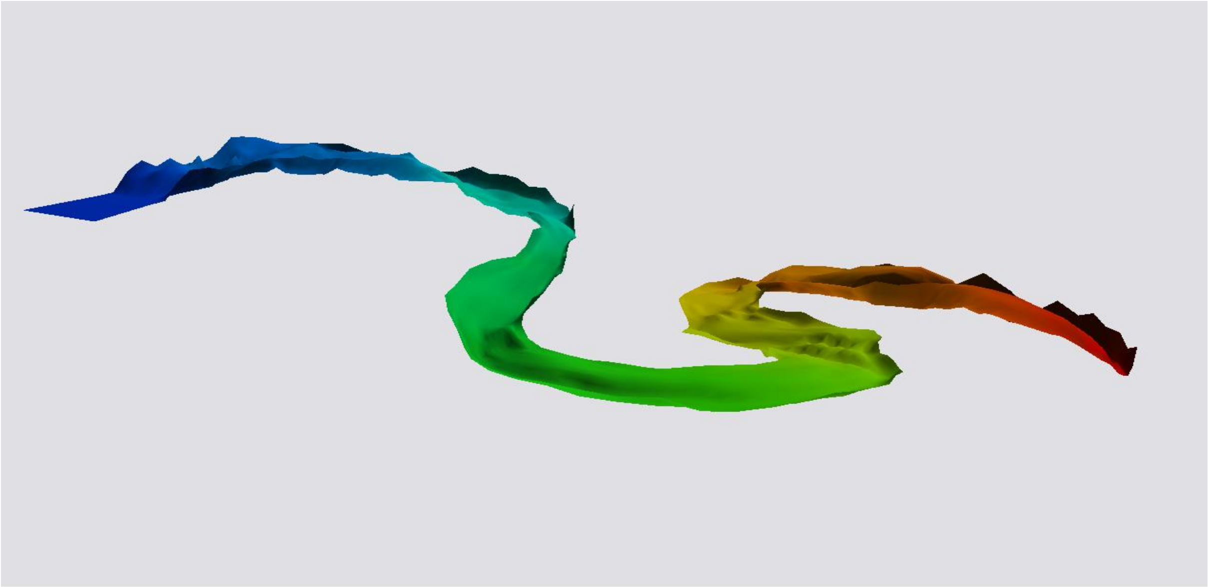
v. 12.3

SMS 12.3 Tutorial

FESWMS – Steering (Incremental Loading)



Objectives

This tutorial demonstrates how to use the steering module with FESWMS to perform automated incremental loading (also known as spinning down or revising the model). The process involves repetitively running the model with boundary conditions getting closer to the desired values. The geometry has already been created and renumbered.

Prerequisites

- Overview Tutorial

Requirements

- FESWMS
- FST2DH
- Mesh Module

Time

- 30–45 minutes

1 Getting Started

Start by opening the geometry file:

1. Select *File* | **Open…** to bring up the *Open* dialog.
2. Browse to the *data files* folder for this tutorial and select the file “Capitol_Reef.fpr”,
3. Click **Open** to import the file and exit the *Open* dialog.
4. If geometry is still open from a previous tutorial, click **Yes** when SMS asks if existing data should be deleted.

The geometry data will open and should appear as shown in Figure 1.

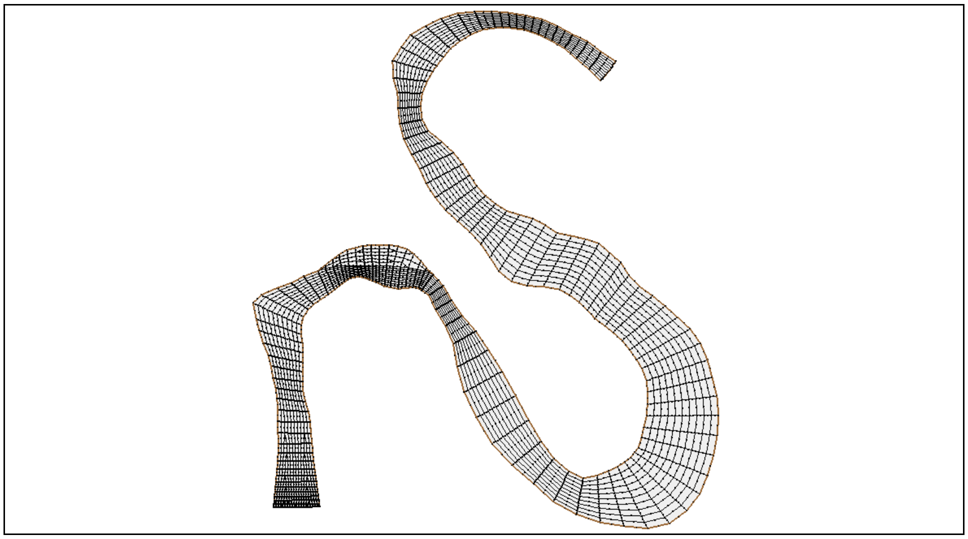


Figure 1 The mesh contained in Capitol_Reef.fpr

 © Aquaveo 2017

2 Specifying Model Units

Set the units and projection as follows:

1. Right-click "Capitol_Reef" in the Project Explorer and select **Projection…** to bring up the *Object Projection* dialog.
2. In the *Horizontal* section, select *No projection.*
3. Turn on *Units* and select "Feet (U.S. Survey)" from the *Units* drop-down.
4. In the *Vertical* section, select "Feet (U.S. Survey)" from the *Units* drop-down.
5. Click **OK** to close the *Object Projection* dialog.
6. Repeat steps 1–5 for " Area Property".
7. Select *Display* | **Projection…** to bring up the *Display Projection* dialog.
8. Repeat steps 2–4.
9. Click **OK** to exit the *Display Projection* dialog.

3 Defining Model Parameters

Several model control parameters must be assigned to define the state of the model. These model parameters include items such as the input and output files, how to handle wetting and drying, the convergence parameters, and the number of iterations to be performed by FESWMS.

Input and output files may also be managed in the *FESWMS Model Control* dialog, including the option to use INI files to hot start the model. This is desirable in complicated networks that require several steps to arrive at a solution. This tutorial uses a different method that automates this process.

Additional information on these parameters and on all of the tabs in the *FESWMS Model Control* dialog is found in the FESWMS Help and the FESWMS documentation.

3.1 General Settings

To define the general model parameters:

1. Select *FESWMS* | **Model Control…** to open the *FESWMS Model Control* dialog.
2. On the *General* tab, enter "Capitol Reef National Park" as the *Network Stamp*.
3. Enter "50 Year Flood" as the *BC Descriptor*.
4. In the *Solution Type* section, select *Steady state*.
5. Continue to the next section without closing the *FESWMS Model Control* dialog.

3.2 Iterations

The *Timing* tab contains options for defining the relaxation factor, number of iterations, and time steps in a dynamic model. The *Relaxation factor* field is an advanced option that can be adjusted to improve how fast the solution will converge. It will not affect the

© Aquaveo 2017

final results. Use the default value, so the only applicable parameter is the number of iterations.

1. On the *Timing* tab, enter "25" for the *Iterations*.
2. Continue to the next section without closing the *FESWMS Model Control* dialog.

3.3 Parameters

The *Parameters* tab is used to set the general parameters of the model, the initial water surface elevation, and the convergence parameters. This tutorial requires changes in only a few of these options.

1. On the *Parameters* tab, in the *General Parameters* section, enter "5070.0 as the *Water-surface elevation*.
2. Enter "1.937" as the *Average water density*.
3. Enter "0.05" as the *Unit flow convergence*.
4. Enter "0.005" as the *Water depth convergence*.
5. Click **OK** to close the *FESWMS Model Control* dialog.

4 Defining Boundary Conditions

For this tutorial, flowrate and water surface elevation will be defined along nodestrings at the open boundaries of the mesh. An open boundary is a boundary where water is allowed to enter or exit. For FESWMS, a flowrate is generally specified across inflow boundaries and water surface elevation is specified across outflow boundaries. Other available boundary conditions are rating curves and reflecting boundaries.

This model has one inflow boundary (bottom left) and one outflow boundary (top right), so two nodestrings must be created. These boundaries are highlighted in Figure 2.

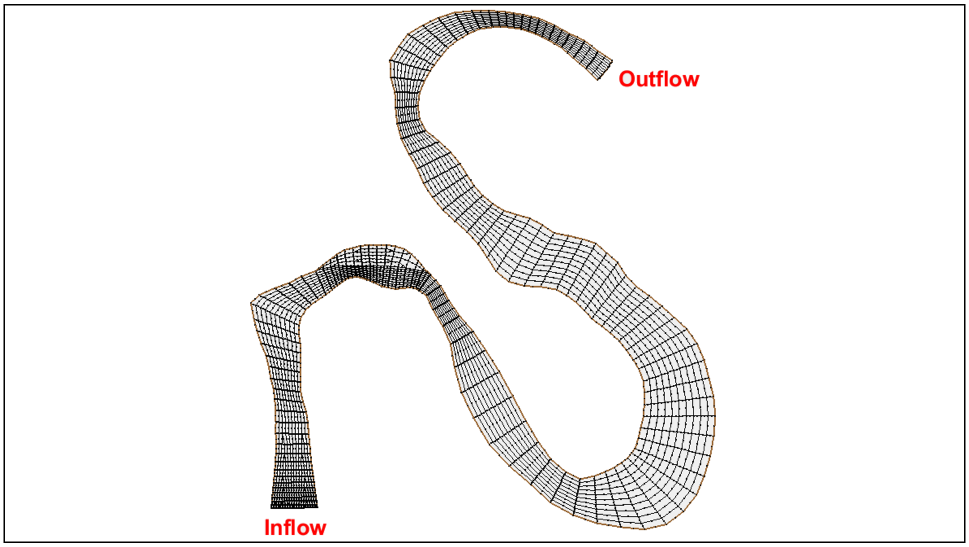


Figure 2 Position of the boundary nodestrings in the mesh

 © Aquaveo 2017

4.1 Creating Nodestrings

Nodestrings should be created from right to left when looking downstream (as if standing in the river and facing downstream) and the first nodestring should be that which spans the whole river section. In this case both nodestrings span the entire river section so it does not matter which nodestring is created first.

To create the outflow nodestring:

1. Select "Capitol_Reef" to make it active.
2. **Zoom** in on the outflow boundary.
3. Using the **Create Nodestring** tool, create the outflow nodestring by clicking on the bottom outlet node (since it would be on the right if facing downstream), then pressing *Shift* while double-clicking on the top outflow node to end the nodestring (Figure 3).

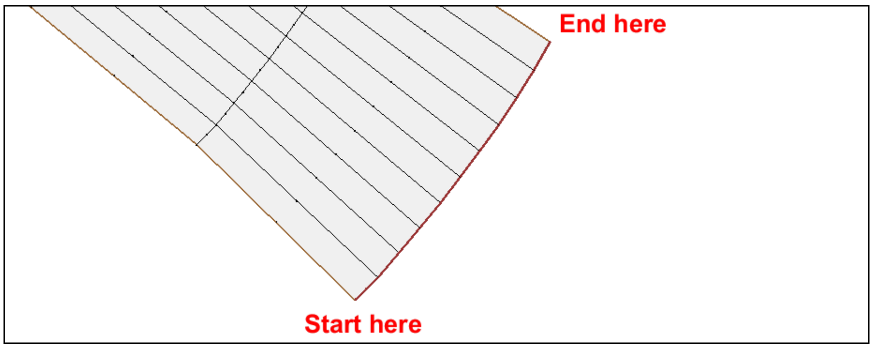


Figure 3 Start and end of the outflow nodestring

4. **Frame** the project and **Zoom** in on the inflow boundary.
5. Repeat step 3, starting with the right node (since it would be on the right if facing downstream) and ending at the left node of the inflow boundary (Figure 4).

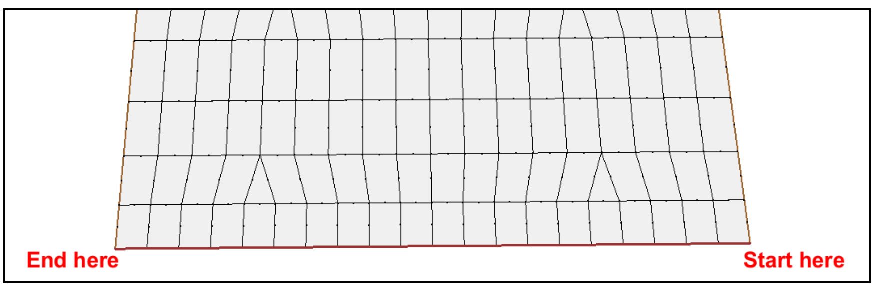


Figure 4 Start and end of the inflow nodestring

4.2 Defining Flow Boundary Conditions

Define the bottom nodestring with an inflow boundary condition as follows:

1. Select "Capitol_Reef" to make it active.
2. Using the **Select Nodestrings** tool, select the inflow nodestring (the one at the bottom of the mesh) by clicking on the square selection icon at the center of the nodestring.

© Aquaveo 2017

3. Select *FESWMS* | **Assign BC…** to bring up the *FESWMS Nodestring Boundary Conditions* dialog.
4. Select "Specified Flow / WSE" from the *Boundary type* drop-down.
5. Turn on *Flow*.
6. Select "Constant" from the first drop-down under *Flow* and "Normal" from the second.
7. Enter "6550.0" as the *Flow rate*.
8. Click **OK** to close the *FESWMS Nodestring Boundary Conditions* dialog.

4.3 Defining the Head Boundary Conditions

A water surface elevation (head) boundary condition will be assigned to the outflow boundary nodestring by doing the following:

1. **Zoom** out from the inflow nodestring so the outflow nodestring is visible.
2. Using the **Select Nodestrings** tool, select the outflow nodestring at the top of the mesh.
3. Right-click and select **Assign BC…** to bring up the *FESWMS Nodestring Boundary Conditions* dialog.
4. Select "Specified Flow / WSE" from the *Boundary type* drop-down.
5. Turn on *Water surface elevation*.
6. Select "Constant" from the first drop-down under *Water surface elevation* and "Essential" from the second.
7. Enter "5070.0" as the *WSE*.
8. Click **OK** to close the *FESWMS Nodestring Boundary Conditions* dialog.

5 Defining Material Properties

Each element in the mesh is assigned a material type ID. This particular geometry has five material types that are viewable by doing the following:

1. Select *Display* | **Display Options** or click **Display Options** to open the *Display Options* dialog.
2. Select "2D Mesh" from the list on the left.
3. On the *2D Mesh* tab, turn on *Materials*.
4. Turn off *Nodes* and *Elements*.
5. Click **OK** to close the *Display Options* dialog.
6. **Frame** the project.

The project should appear similar to Figure 5 (though the colors and patterns for each material may be different). Most of the model is made of brush floodplain and the channel, but there are a few elements with other material types.

 © Aquaveo 2017

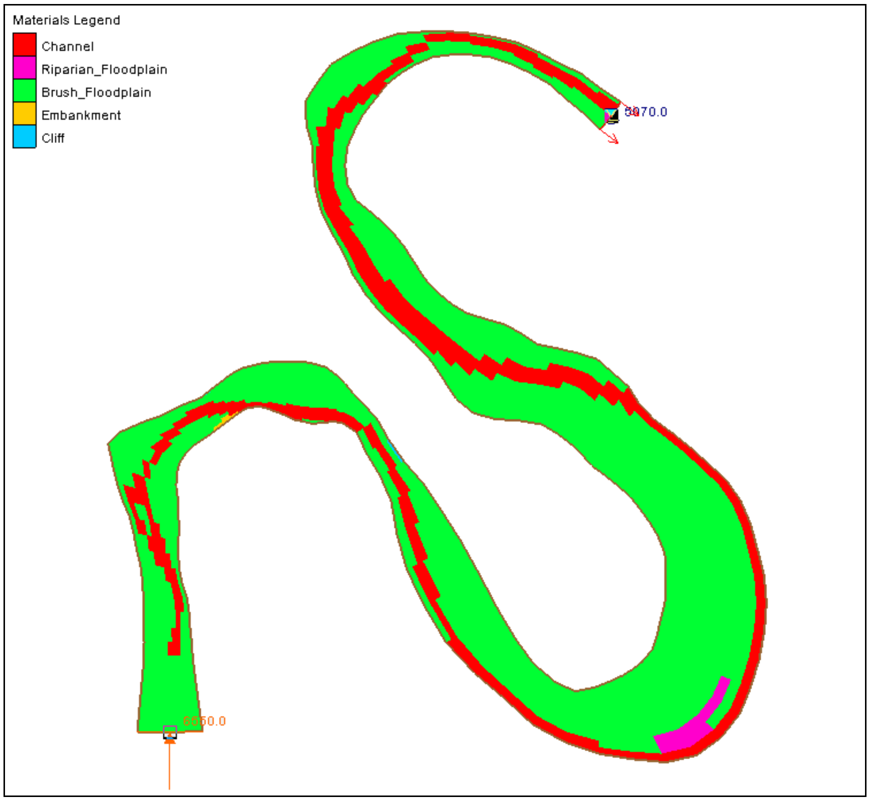


Figure 5 The display of materials

Before continuing, turn off the material display by doing the following:

1. Click **Display Options** to open the *Display Options* dialog.
2. Select “2D Mesh” from the list on the left.
3. On the *2D Mesh* tab, turn off *Materials* and turn on *Elements*.
4. Click **OK** to close the *Display Options* dialog.

The materials were created with default parameters that must be changed for this simulation. The material properties define how water flows through the element.

To define the eddy viscosity and roughness parameters for this model:

1. Select *FESWMS* | **Material Properties…** to bring up the *FESWMS Materials Properties* dialog.
2. Select “Brush Floodplain” from the list on the left.
3. Using the table below, set the roughness (both *n1* and *n2*) on the *Roughness Parameters* tab
4. Using the following table, set the eddy viscosity (*Vo*) on the *Turbulence Parameters* tab.
5. Repeat steps 2–4 for each of the other materials (Channel, Riparian_Floodplain, Embankment, and Cliff) using the table below.
6. Click **OK** to close the *FESWMS Materials Properties* dialog.

 © Aquaveo 2017

Material	***Roughness Parameters*, *n1* and *n2***	***Turbulence Parameters*, *Vo***
Brush_Floodplain	0.05	20
Channel	0.03	20
Cliff	0.05	20
Embankment	0.04	20
Riparian_Floodplain	0.1	20

6 Saving the Simulation

Before running the FST2DH engine, the data must be saved as an SMS project or as an FST2DH simulation. Saving an SMS project file is generally preferred because it will store the scatter data, map coverages, and display settings as well as the data specifically used by FST2DH.

To save the SMS project:

1. Select *File* | **Save As…** to open the *Save As* dialog.
2. Select "Project Files (*.sms)" for *Save as type*.
3. Enter "CR_Sim.sms" as the *File name*.
4. Click **Save** to save the project under the new name and close the *Save As* dialog.

7 Running the Model

To run FST2DH:

1. Select *FESWMS* | **Run FST2DH**.

Before SMS launches the model, a quick check is done on the data to make sure everything is valid. This model check will bring up the *Model Checker* dialog shown in Figure 6 if any anomalies are detected. For this model, three warnings should be detected.

The first warning says that the elements might dry out (Figure 6), so the wet/dry flag should be turned on by doing the following:

2. Click **Cancel** to leave the *Model Checker* dialog without running *FST2DH*.
3. Select *FESWMS* | **Model Control…** to open the *FESWMS Model Control* dialog.
4. On the *Parameters* tab, in the *General Parameters* section, enter "0.1" as the *Default storativity depth (ft)*.
5. Turn on *Element drying / wetting*.
6. Enter "0.1" as the *Depth tolerance for drying (ft)*.
7. Click **OK** to close the *FESWMS Model Control* dialog.
8. Resave the project (*Ctrl-S*).

© Aquaveo 2017

9. Select *FESWMS* | **Run FST2DH** to launch the *Model Checker* and verify the first error has gone away.

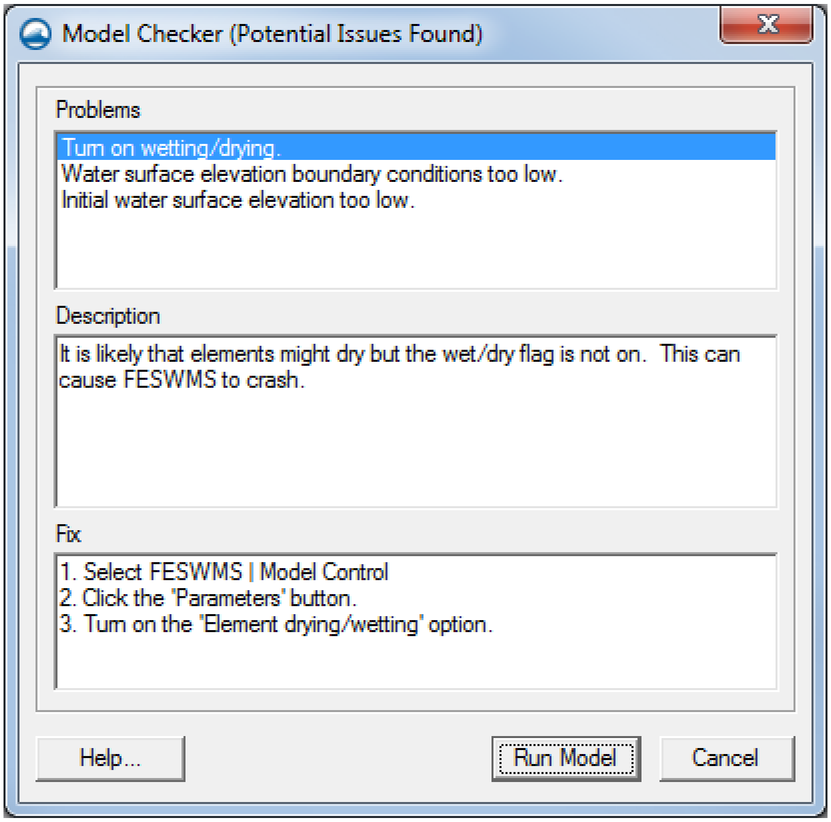


Figure 6 Warning in FESWMS data

The remaining warnings say that the initial water surface elevation and the WSE boundary condition for the simulation are too low and will leave portions of the domain dry. This can lead to instabilities and cause FST2DH to crash.

One option is to raise the initial water surface elevation, but if it is much higher than the outflow boundary elevation, instabilities can develop at those locations. If the simulation is not stable at the WSE boundary condition, it may be best to use incremental loading (as will be demonstrated later).

For now, ignore these warnings. Because there is still a warning message, FST2DH might not converge. However, before attempting to fix this, the simulation will be tried.

10. Click **Run Model** to launch FST2DH.

FST2DH will run iteration 1 and diverge. Near the bottom of the *FESWMS Output* section in the *FESWMS* dialog is the text "***Run ended abnormally because of 1 errors and 0 warnings!" as shown in Figure 7. This is how FST2DH declares that it has not successfully converged.

11. Click the **Exit** button to close the *FESWMS* window.

Various things can contribute to a model not converging. In this case, SMS gave an error message that the initial water surface was too low. The low water surface elevation for this simulation does not allow FST2DH to converge from the cold start simulation.

 © Aquaveo 2017

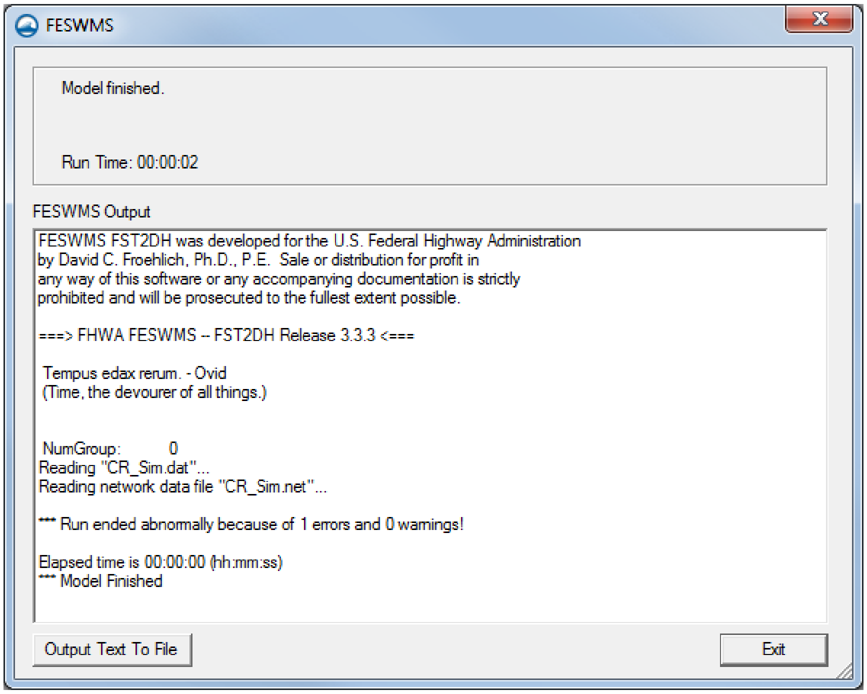


Figure 7 Output from running CR_Sim.fpr

To illustrate why this occurs, compare the boundary condition with the bathymetry:

1. Select *File* | **Get Info**—or click **Get Module Info** —to bring up the *Information* dialog.
2. On the *Mesh Module* tab, look at the *Maximum Z value* (highlighted in Figure 8).

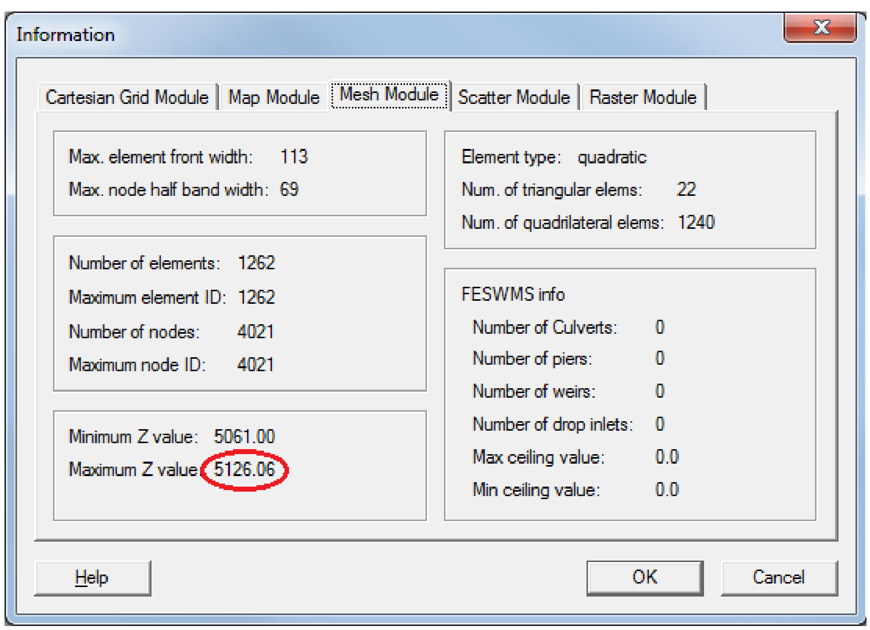


Figure 8 Mesh Information dialog with Maximum Z value highlighted

The maximum bathymetry elevation is well above the boundary condition of 5,070 ft. This means not all of the nodes are wet with the initial water surface elevation. FST2DH

 © Aquaveo 2017

requires that all nodes be wet for the initial condition or the model will not run. This is why FST2DH diverged after only running the first iteration.

3. Click **OK** to close the *Information* dialog.

8 Using the Steering Module

It is possible to manually set the boundary water surface elevation high enough to wet all nodes. The model could then be run and an output solution file could be created. This solution file could be used to hot start the model with a lower boundary water surface elevation. This manual process could then be repeated until the boundary water surface elevation is at the desired level. However, that would require an enormous amount of user input and time, especially for this model.

This process can be automated by using the steering module by doing the following:

1. Select *Data* | **Steering Module** to bring up the *Steering Wizard* dialog.
2. Select *FESWMS Spindown* and click **Next** to go to the *FESWMS Steering* page of the *Steering Wizard* dialog.
3. Turn on *Delete intermediate steering files*.
4. In the *Spindown* section, turn on *Water Surface Elevation* and turn off all other options in the *Spindown* section (Figure 9).
5. Click **Start** to close the *Steering Wizard* dialog and bring up the *FEWMS Spindown* dialog.

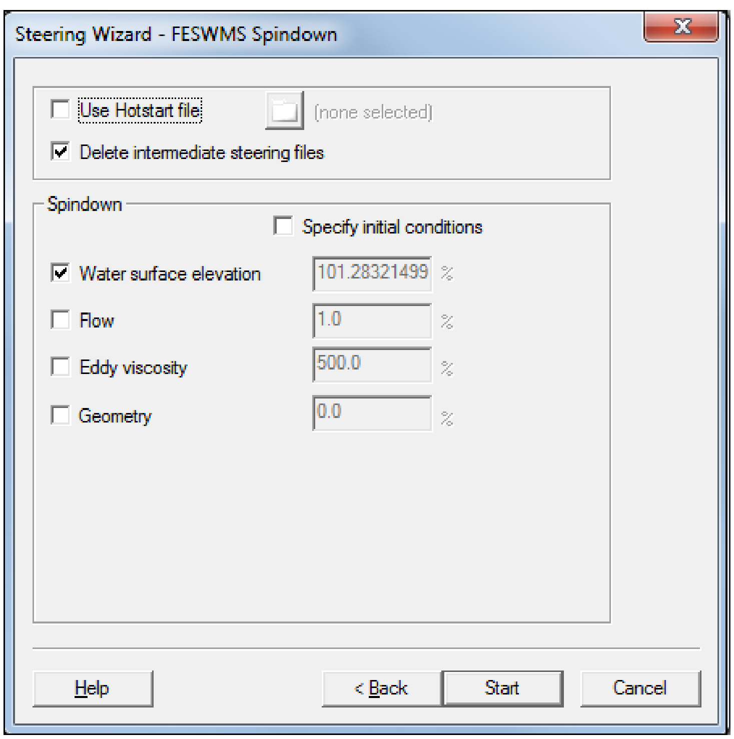


Figure 9 Steering Module dialog

 © Aquaveo 2017

The *FESWMS Spindown* dialog (Figure 10) updates with the progress of spinning down the model. The top window explains the WSE convergence of the current run. Each iteration shows as a green point, allowing determination of if the run is converging or diverging (moving toward or away from 0 head change). The iteration being performed is shown just above this plot.

The bottom window shows the overall spindown of the model. The green points represent successful runs, and the red Xs represent failed runs. When this plot reaches 100% spun down, the model is finished. This percent is shown just above this plot.

This process can take several minutes to complete. When the spindown has finished, a warning window appears advising that the "Steering process has terminated – See status file for details".

6. Click **OK** to close the completion message.
7. Click **OK** to close the *FESWMS Spindown* dialog.

This status file is named "SteeringStatus.txt" and gives a summary of the steering process. A final solution file has also been created.

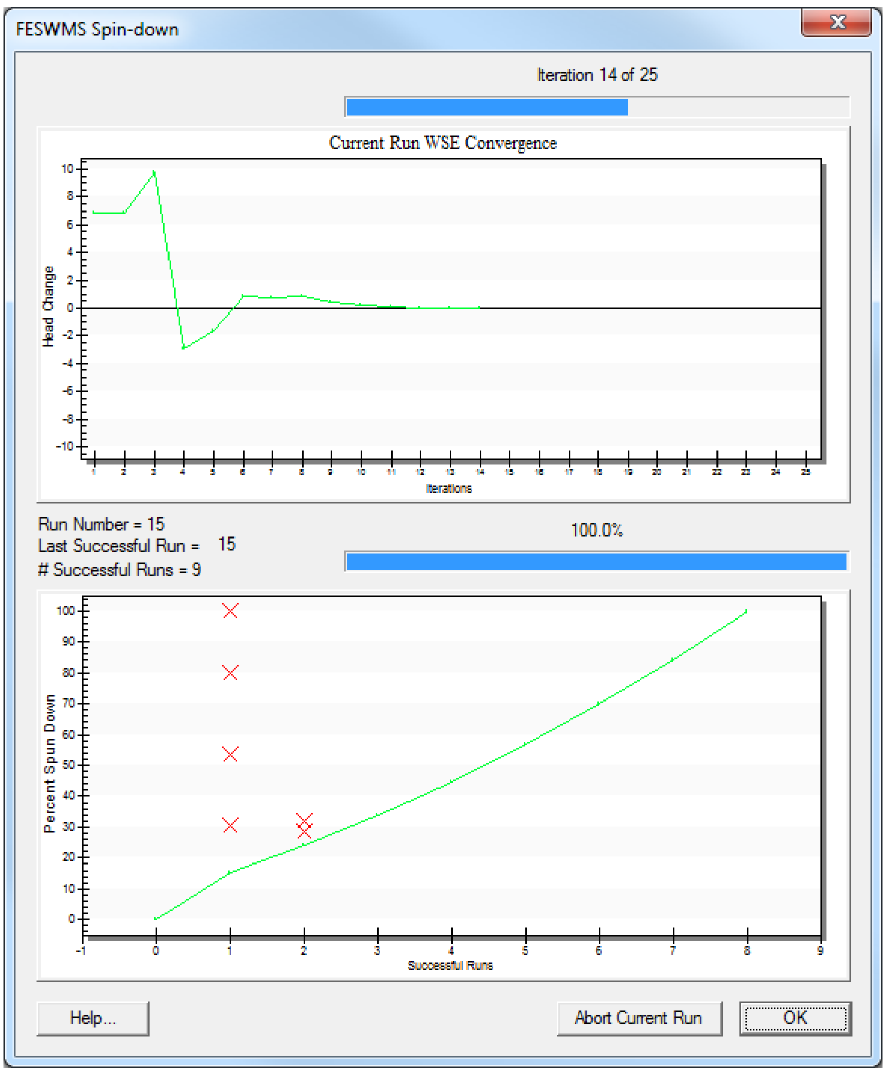


Figure 10 FESWMS Spindown dialog

 © Aquaveo 2017

9 Opening the Solution

To view the solution file:

1. Select *File* | **Open…** to bring up the *Open* dialog.
2. Browse to the *data files* folder and select "Capitol_Reef.flo".
3. Click **Open** to import the FLO file and exit the *Open* dialog.

This adds a new mesh folder named " Capitol_Reef.flo (FESWMS)" under the " Capitol Reef" mesh in the Project Explorer.

4. Select *Display* | **Display Options…** to open the *Display Options* dialog.
5. Select "2D Mesh" from the list on the left.
6. On the *2D Mesh* tab, turn on *Contours* and *Vectors.*
7. On the *Contours* tab, in the *Contour method* section, select "Color Fill" from the first drop-down.
8. On the *Vectors* tab, in the *Arrow Options* section, set *Shaft length* to "Define min. and max. length".
9. Set the *Minimum* to "5" and *Maximum* to "20".
10. Click **OK** to exit the *Display Options* dialog.

The project should appear similar to Figure 11.

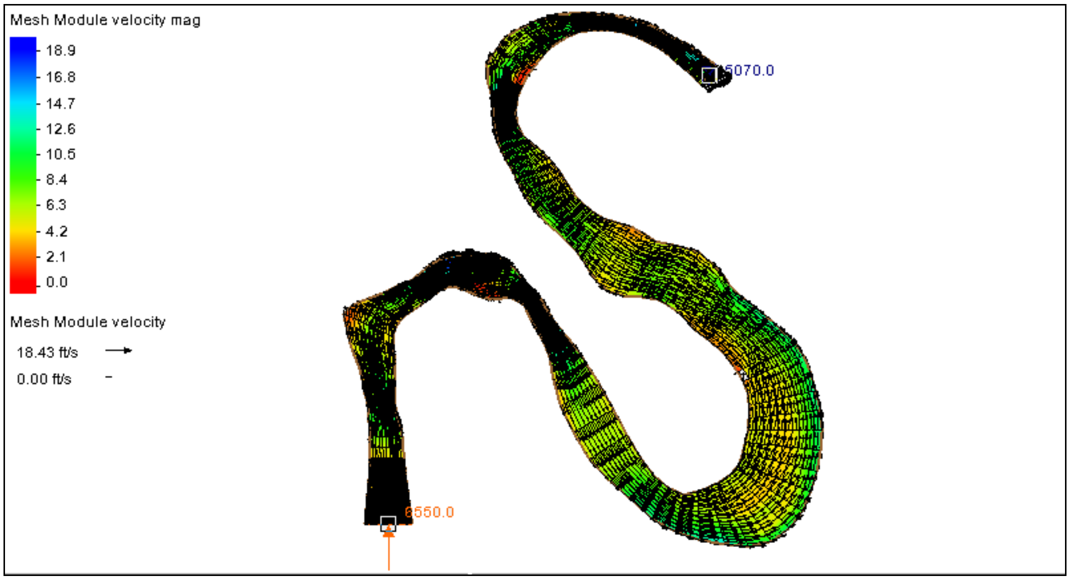


Figure 11 Velocity vectors and contours visible

10 Conclusion

This concludes the "FESWMS Steering (Incremental Loading)" tutorial. Continue reviewing and post-processing the results if desired. When finished, exit SMS.

© Aquaveo 2017